ECAD Import Module

User's Guide

5.5

COMSOL

ECAD Import Module User's Guide

Version: COMSOL 5.5

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM023601

Contents

Chapter 1: Introduction

Chapter 2: Importing ECAD Files

Chapter 3: Programming and Command Reference

1

Introduction

Welcome to the ECAD Import Module! This *User's Guide* details features and functionality available with the product.

This introductory chapter is About This Guide.

About This Guide

This documentation covers the ECAD Import Module.

About the ECAD Import Module

This product extends the functionality of COMSOL Multiphysics by enabling modeling using imported ECAD design files.

The import capabilities cover the *GDS II*, *NETEX-G*, *ODB++®,* and *ODB++(X)* file formats and include the creation of 3D geometry during the import operation.

Where Do I Access the Documentation and Application Libraries?

A number of internet resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or context-based) help, and the application libraries are all accessed through the COMSOL Desktop.

> **!** If you are reading the documentation as a PDF file on your computer, the blue links do not work to open an application or content referenced in a different guide. However, if you are using the Help system in COMSOL Multiphysics, these links work to other modules (as long as you have a license), application examples, and documentation sets.

THE DOCUMENTATION AND ONLINE HELP

The *COMSOL Multiphysics Reference Manual* describes all core physics interfaces and functionality included with the COMSOL Multiphysics license. This book also has instructions about how to use COMSOL Multiphysics and how to access the electronic Documentation and Help content.

Opening Topic-Based Help

The Help window is useful as it is connected to many of the features on the GUI. To learn more about a node in the Model Builder, or a window on the Desktop, click to highlight a node or window, then press F1 to open the Help window, which then displays information

about that feature (or click a node in the Model Builder followed by the **Help** button (?). This is called *topic-based* (or *context) help.*

	To open the **Help** window:
Win	• In the **Model Builder**, **Application Builder**, or **Physics Builder** click a node or window and then press F1.
	• On any toolbar (for example, **Home**, **Definitions**, or **Geometry**), hover the mouse over a button (for example, **Add Physics** or **Build All**) and then press F1.
	• From the **File** menu, click **Help** (?).
	• In the upper-right corner of the COMSOL Desktop, click the **Help**(?) button.

	To open the **Help** window:
Mac	• In the **Model Builder** or **Physics Builder** click a node or window and then press F1.
Linux	• On the main toolbar, click the **Help** (?) button.
	• From the main menu, select **Help>Help**.

Opening the Documentation Window

Win	To open the **Documentation** window: • Press Ctrl+F1. • From the **File** menu select **Help>Documentation** (▊).

Mac **Linux**	To open the **Documentation** window: • Press Ctrl+F1. • On the main toolbar, click the **Documentation** (▊) button. • From the main menu, select **Help>Documentation**.

THE APPLICATION LIBRARIES WINDOW

Each application includes documentation with the theoretical background and step-by-step instructions to create a model application. The applications are available in COMSOL as MPH-files that you can open for further investigation. You can use the step-by-step instructions and the actual applications as a template for your own modeling and applications. In most models, SI units are used to describe the relevant properties, parameters, and dimensions in most examples, but other unit systems are available.

Once the Application Libraries window is opened, you can search by name or browse under a module folder name. Click to view a summary of the application and its properties, including options to open it or a PDF document.

🔍	The Application Libraries Window in the *COMSOL Multiphysics Reference Manual*.

Opening the Application Libraries Window
To open the **Application Libraries** window (▦):

Win	• From the **Home** toolbar, **Windows** menu, click (▦) **Applications Libraries**. • From the **File** menu select **Application Libraries**. To include the latest versions of model examples, from the **File>Help** menu, select (▦) **Update COMSOL Application Library**.

Mac **Linux**	Select **Application Libraries** from the main **File>** or **Windows>** menus. To include the latest versions of model examples, from the **Help** menu select (⊞) **Update COMSOL Application Library**.

CONTACTING COMSOL BY EMAIL

For general product information, contact COMSOL at info@comsol.com.

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and case number is sent to you by email.

COMSOL WEBSITES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
Support Center	www.comsol.com/support
Product Download	www.comsol.com/product-download
Product Updates	www.comsol.com/support/updates
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/community
Events	www.comsol.com/events
COMSOL Video Gallery	www.comsol.com/video
Support Knowledge Base	www.comsol.com/support/knowledgebase

2

Importing ECAD Files

ECAD file formats have been designed to transfer information necessary for the manufacturing of printed circuit boards (PCBs), integrated circuits (ICs) and microelectromechanical systems (MEMS) devices. You can import data from ECAD files to generate geometry for simulation. During the import of an ECAD file geometry objects are generated based on the 2D layouts and stackup information found in the files.

In this section:

- Importing IPC-2581 and ODB++ Files
- Importing GDS-II Files
- Meshing an Imported Geometry
- Troubleshooting ECAD Import

Importing IPC-2581 and ODB++ Files

In this section:

- File Formats for Printed Circuit Boards (PCBs)
- Extruding Layers
- Import Settings for IPC-2581 and ODB++ Files

File Formats for Printed Circuit Boards (PCBs)

The IPC-2581 and OBD++ file formats can handle most of the information needed to manufacture a PCB. For generating a geometry for simulation only some of this information is needed, such as the layout of copper layers, the board outline, and layer stackup information.

An ECAD file for PCBs may also include layers with 2D layouts that specify, for example, the vias and component outlines on the board. You can also import these layers. In a file, the 2D layouts consist of shapes, also called symbols that build the geometry of copper traces and pads. A large number of symbols are specified by the supported PCB formats, for example, lines, circles, rectangles, and surfaces to name a few.

THE ODB++ FORMAT

The ODB++ format defines a directory structure with several files, each containing parts of information about the PCB. An entire PCB layout is often distributed as zipped or unzipped tar archives.

The ODB++ import reads the layer list and the first step in the file. From the first step the layer features and the board outline are extracted, but the package information is skipped. Multiple step files are not supported.

	Microwave Filter on PCB with Stress: Application Library path **RF_Module/Filters/pcb_microwave_filter_with_stress**

Extruding Layers

To obtain a 3D geometry for the board the 2D layouts from the PCB file are extruded, according to the layer stackup information, using one of two methods. A special extrude operation can automatically connect the imported layers to generate a single geometry object after the import. The extruded layers correspond to domains in the resulting object. If the 2D layouts are such that the special extrude algorithm cannot be used a second method is available that performs several extrude operations to generate one geometry object per layer. See how to select the import method in Grouping of Geometries.

Import Settings for IPC-2581 and ODB++ Files

To add an **Import** node, from the **Home** or **Geometry** toolbar, click **Import** (![icon]). In the **Import** section of the Settings window, set the type of file to import to **ECAD file**. You can also skip this step as the type of the selected file is automatically recognized by the code. Click **Browse** to locate the file to import, or enter the path to the file. Before clicking the **Import** button consider to review and configure the import options, especially the layer thickness information, since in many cases the IPC-2581 files and ODB++ archives do not include all necessary information to construct a 3D geometry.

The length unit in the file is detected and displayed in the Settings window. To use the unit in the file as the length unit for the geometry sequence select the **Update geometry unit** check box. The check box is selected by default if the **Import** node is added as the first node in the geometry sequence. The length unit of the geometry sequence is set to `inch` when the length unit `mil` is detected in the selected file.

GROUPING OF GEOMETRIES

To determine the extrusion method used for generating the 3D geometry from the 2D layouts select one of the available alternatives from the **Grouping of geometries** list:

- **All**. This alternative is available only in 3D. The import algorithm extrudes in one operation all imported copper and dielectric layers into a single 3D geometry object. From each copper layer, the imported symbols, such as lines, pads, and surfaces, are combined before extrusion. The copper and dielectric layers become domains in the resulting object. To be able to connect the layers, the algorithm requires that the 2D layouts fulfill certain rules. If the import fails, switch to the **By layer** grouping option.

- **By layer**. Use this alternative to extrude each layer separately. The imported symbols, such as lines, pads, or surfaces, from the copper layers are combined before

extrusion. Several geometry objects, one for each imported copper and dielectric layer, are output by the import.

- **No grouping**. Use this alternative to import and extrude individually each symbol from the copper layers. The import results in separate objects for imported symbols and dielectric layers.

When using the **No grouping** alternative you have the option to import line symbols as curve objects by selecting the **Ignore line width** check box.

Note: Using the ignore line width option you can construct geometry objects for components on the board from the information on layers with component outlines. To do this configure an Import node to import only the component outline layer. Select the **No grouping** method, and the **Ignore line width** check box. Also select the **Manual control of elevations** check box, and set the elevation to an appropriate value, but keep the default zero layer thickness. After the import use geometry operations to convert the component outline curve objects to surface objects, and extrude those to 3D components.

COPPER LAYERS

The **Type of import** list specifies how to treat copper layers. The **Full 3D** option imports all copper layers with a thickness according to the **Layers to import** table. With the **Metal shell** options copper layers are imported as embedded boundaries between dielectric regions. As illustrated in the figure below, this setting does not affect the thickness and

elevation of the dielectric layers, which are always extruded according to the data in the **Layers to import** table.
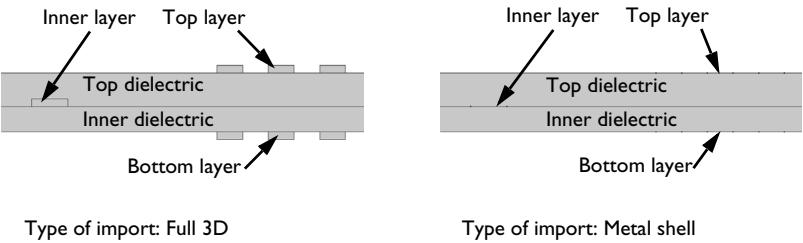
Figure 2-1: Cross-section of a PCB with three copper layers after Full 3D import, to the left, and Metal shell import, to the right.

For 3D imports, when the geometry grouping **All** and the import type **Metal shells** are used together, an isolated copper layer cannot be imported if the import also includes another solid (dielectric) layer. In this case use the **By layer** grouping, or add two Import features to the geometry sequence, one to import the copper layer, and a second one for the solid layers.

CONTROLLING THE LAYER STACKUP

The import can read stackup information from the PCB file, such as the thickness for copper and dielectric layers. The layer information is displayed in the **Layers to import** table. Sometimes the layer thickness is not included in the export from the ECAD program, so the layers get the default zero thickness. Therefore it is recommended that these values are checked before importing.

Select the **Manual control of elevations** check box to enable the repositioning of layers in the stackup. This check box is enabled when **Grouping of geometries** is set to **By layer** or **No grouping**. When the **Manual control of elevations** check box is cleared, the z positions of the layers are calculated automatically from the layer **Thickness** values.

Select the **Show names from file** check box to display the layer names as included in the PCB file. This is useful after assigning new layer names that are used in the COMSOL application.

The **Layers to import** table contains the following columns:

- The **Name** column displays the layer names from the file. You can edit the entries to give more descriptive names to the layers. The names that appear here are used to name the resulting geometry objects and the selections when the **Layer selections** check box is selected under **Selections of Resulting Entities**.

- The **Name in file** column is visible only when you select the **Show names from file** check box above the table. The **Name in file** column always displays the layer name from the PCB file, even after editing the **Name** column.

- The **Type** column declares the type of layers. Depending on their type, the import treats layers differently during import. For example, the geometry from a layer of type **Metal** is extruded only if the **Type of import** is set to **Full 3D**. A union of the objects from the **Outline** layer is used as the PCB outline to determine the extent of the dielectric layers. The **Drill** layer type means that the objects in the layer define drilled via holes through the PCB.

- The values in the **Thickness** column are used as the extrusion distances for the layers. Layer elevations are also calculated based on these values when **Manual control of elevations** is not enabled. The values in the layer **Thickness** column can always be changed prior to import.

- The values in the **Elevation** column control the lower z positions of the layers. By adjusting the values you can adjust the position of layers in the stackup. The **Elevation** column is only displayed when **Manual control of elevations** is enabled.

- The **Import** column. Clear the check box for layers that do not need to be imported.

To save the layers table to a text file, click the **Save to File** button (⊟) under the table. To load layer table information from a text file, click the **Load from File** button (▶) under the table. Only the editable data in the table is affected when loading from file. Layer data in the file that does not match the existing layers in the table is ignored. To reset the thickness and elevation values in the table to the last read data from the PCB file click the **Reset Thickness and Elevation** button (↻). To select the **Import** check box in the table for all copper and dielectric layers click the **Select All Metal and Dielectric Layers for Import** button (▯). Click the **Clear All Imports** button (▯) to clear all check boxes in the **Import** column. To reload the layer information from the PCB file to the table click the **Reload Layers** button (▤).

DIELECTRIC LAYERS

When the PCB file includes the outline of the PCB board in the file the dielectric regions are generated to the shape of the board. Even if the dielectric layers are selected

for import in the **Layers to import** table, the import and extrusion of the corresponding objects or domains can be turned off by clearing the **Import dielectric regions** check box.

INTERIOR BOUNDARIES

By default the **Keep interior boundaries** check box is cleared to instruct the import to remove all interior boundaries on the imported layers. Interior boundaries on the copper layers are the result of the union of the individual symbols that make up the traces. Ignoring the interior boundaries keeps the geometry complexity to a minimum and can also make the import more robust in some situations.

COPPER TEXT AND OBJECTS OUTSIDE THE BOARD BOUNDARY

It is not uncommon that the copper layouts have text objects that provide information, and are not part of any electrical circuits. Shapes may be also included for example to mark the board boundaries, and these shapes are often positioned outside the board. Such objects may make it more difficult to construct the geometry and are usually of no interest in a physical simulation.

Select the **Ignore text objects** check box to skip all objects in a PCB file that have the TEXT tag set. To prevent objects outside the board from being imported select the check box **Ignore objects outside of board**. Objects that are positioned outside the board, at a distance greater than that specified by the offset in the **Offset from boundary** edit field, are ignored by the import. The default value for the offset is -0.1 mm, which means that objects that are further into the board than 0.1 mm from the board boundary are imported. The offset should only be taken as an approximation. It is only checked where the object has its defining points, which are not necessarily the same points that can seen on an object after it has been imported. For example, a trace is defined by its start and end points, which lie on the center line of the trace. Similarly, a circle might be defined by four points (south, north, east and west). Thus, an object might be ignored even if an edge is further inside the board than the specified by the offset, as long as all the points that define the object are further away from the boundary than the offset.

REPAIR

Geometry repair is controlled via the **Repair imported data** check box and the **Repair tolerance** list. Change the **Repair tolerance** from **Automatic** to **Relative** to manually specify a tolerance in the **Relative repair tolerance** edit field. **Automatic** means a relative repair tolerance of 10^{-6}.

When selected, the geometry repair is active when combining the objects to generate the geometry of each layer and also when the layers are combined together. By increasing the repair tolerance, problems with short edges arising when combining the objects can sometimes be circumvented. A repair tolerance that is too high may however lead to breaking the geometry. As a guideline, the relative tolerance should be kept between 10^{-5} to 10^{-8}. Values at the lower end of this interval may result in faster import at the cost of a higher number of short edges being present in the imported geometry. Note that after the import short edges can automatically be removed from the geometry using the Remove Details feature, see Remove Details in the *COMSOL Multiphysics Reference Manual*. For an example of using the Remove Details feature on imported PCB geometry see the tutorial *Importing and Meshing a PCB Geometry from an ODB++ Archive* found in the Application Libraries for the ECAD Import Module.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** check box to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Select the **Layer selections** check box to create predefined selections — for domains, boundaries, and objects — in subsequent nodes in the geometry sequence for each imported layer in the PCB file. The boundary selection for drill layers include all boundaries of the objects, whereas for all other layer types only the horizontal boundaries are included in the selection.

To make the resulting entities that the layers consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an

option from the **Show in physics** list: **All levels**, **Domain selection**, or **Boundary selection**. The default is **All levels**, which makes predefined selections available on all applicable levels, suitable for use with materials and physics defined in domains and boundaries. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

NET SELECTIONS

The sections **Domain Net Selections** (in 2D and 3D) and **Boundary Net Selections** (in 3D), contain a table that lists the generated selections for the imported nets. Net selections are generated when importing IPC-2581 and ODB++ files. The tables contain the following columns, ordered from left to right:

- **Name** - This column contains the name of the selection.

- **Keep** - The check box in this column is cleared by default, and determines whether the selection will be available for use in geometry features following the **Import** feature.

- **Physics** - The check box in this column is selected by default. Provided that also the **Keep** check box has been selected, it determines whether the selection will be available in all applicable selection lists in physics and materials settings, for example. If the Import node has been added under the Plane Geometry node of a Work Plane, this column header is titled **3D**. If the Import node has been added to a geometry part, this column header is titled **Instances**.

- **Contribute to** - Select a cumulative selection from the list in this column to make the resulting entities contribute to a cumulative selection. The default, **None**, gives no contribution. Contributing to a cumulative selection does not require that the **Keep** check box has been selected.

Click the **New Cumulative Selection** button under the tables to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

The entities in a selection are highlighted in yellow in the **Graphics** window when the selection is clicked in the table. To zoom in on the selection use the **Zoom to Selection** button in the **Graphics** toolbar.

The net selection names are derived from the net and layer names imported from the PCB file. For example, for a net with the name NET on a board with three copper layers (SIG1, GND, SIG2), two dielectric layers (SIG1.DIEL, GND.DIEL), and two drill layers (DRILL1, DRILL2), the following selections are generated: NET.SIG1, NET.GND,

NET.SIG2, NET.DRILL1_SIG1_DIEL, NET.DRILL1_GND_DIEL, NET.DRILL2_SIG1_DIEL, NET.DRILL2_GND_DIEL, and NET. The selection NET is the union of all other selections for this net. If the name of a net is the same as the name of a layer, for example GND, the suffix _N is attached to the name of the net selection to avoid a name clash between net and layer selections.

Empty net selections are not generated, i.e. a selection for the net is generated only when the intersection of the net and the layer contains entities. For entities that do not belong to a net the selection NONET is generated.

The boundary net selections include only the horizontal boundaries for the copper and dielectric layers, and both vertical and horizontal boundaries for the drill layers. Note that edge and vertex selections are not generated for nets.

Importing GDS-II Files

In this section:

- The GDS-II File Format
- Extruding Layers
- Import Settings for GDS-II Files

The GDS-II File Format

The GDS-II file format is commonly used for mask layout production used in the manufacturing process of semiconductor devices and MEMS devices. The file is a binary file, containing information about drawing units, geometry objects made of polygons, and object drawing hierarchy. The drawing hierarchy is made up of a library of cell definitions, where each cell can be instantiated (drawn several times) with scaling, translation, mirroring, and rotation. It is also possible to repeat a cell as an array of drawn objects. This is very useful for mask layouts of integrated circuits, which often consist of millions of transistors. There are usually only a few transistor configurations present on the layout, and each transistor configuration only has to be defined once. You can configure the import to include only selected cells, see Cell Selection. Geometry objects in the cells belong to different layers which represent different steps in the manufacturing process. Geometry objects can also be assigned data types that are sometimes used group together objects.

FILE EXTENSION

The file extension of the GDS-II format is usually `.gds`, and the ECAD import requires it to be so, otherwise it cannot identify the file as a GDS-II file. If the file has a different extension, it must be changed to `.gds` before importing the file.

SUPPORTED FEATURES

There are several record types in a GDS-II file that are of no interest in a geometry import and these are ignored. There are also a few record types that actually could be imported as a geometry object, but are also ignored. One such example is the Text record, which produce a lot of mesh elements and is usually of no interest in a simulation. Below is a list of the supported record types:

- Boundary: a closed polyline object
- Box: a box object

- Path: a path with a thickness
- Sref: an instance of a cell that can be translated, rotated, scaled, and mirrored
- Aref: an n-by-m array of Sref objects
- Element: specification of a cell

Extruding Layers

The GDS-II format does not contain layer stackup information, such as thickness and elevation, this has to be supplied before import to create a 3D structure by extrusion. For extruding the layers, the import algorithm uses two methods. A special extrude operation can automatically connect the imported layers to generate a single geometry object after the import. The extruded layers correspond to domains in the resulting object. If the geometry on the layers is such that the special extrude algorithm cannot be used a second method is available that performs several extrude operations to generate one geometry object per layer.

Several layers on the same height is common for semiconductor layouts, where the fabrication process includes deposition followed by etching and then redepositing of a different layer. To handle such a process scheme the method of separate extrusions for each layer is necessary. After the import, you can do etching by removing a layer from other objects, by using the **Difference** button from the **Boolean Operations** submenu on the Geometry toolbar. You can find the various import options described under the section Grouping of Geometries.

Import Settings for GDS-II Files

To add an **Import** node, from the **Home** or **Geometry** toolbar, click **Import** ( ). In the **Import** section of the Settings window, set the type of file to import to **ECAD file**. You can also skip this step as the type of the selected file is automatically recognized by the code. Click **Browse** to locate the file to import, or enter the path to the file. Before clicking the **Import** button configure the layer thickness, since the GDS-II files do not include the necessary layer thickness and elevation information to construct a 3D geometry.

For the length unit in the file μm is always displayed, since GDS-II files do not specify a unit. To use the displayed unit as the length unit for the geometry sequence select the **Update geometry unit** check box. The check box is selected by default if the **Import** node is added as the first node in the geometry sequence.

CELL SELECTION

Data contained in GDS-II files is organized in hierarchical structure consisting of cells. Select a *cell* name from the **Cell to import** list to limit the import to the selected cell beneath the top cell in the hierarchy. Use the default choice in this list to import the top cell. By default all subcells below the selected cell are imported. To import only one sub-cell type, select the cell from the **Filter by subcell** list. The difference between selecting a cell and filtering the top cell by the same cell is that the latter alternative includes all instances of the selected cell, whereas the first case only includes the selected instance of the cell.

GROUPING OF GEOMETRIES

The import operation can construct a single 3D geometry object with separate domains for the layers and the various objects on the layers. You also have the option to import each layer into its own geometry object, or to import each object from the layers in the file into its own geometry object. To determine which method to use select one of the available alternatives from the **Grouping of geometries** list:

- **All**. This alternative is available only in 3D. Use it to group all imported layers and the object contained on those into a single geometry object. To construct the 3D geometry object the import algorithm extrudes and combines all layers directly. The imported objects, and layers become domains in the resulting geometry object.

- **By layer**. Use this alternative to combine the imported objects from each layer into a single geometry object. Several geometry objects, one for each imported layer, are output by the import.

- **No grouping**. Use this alternative to import each object from the layers into individual geometry objects. To construct the geometry objects the import performs a union of all the segments.

With the **Type of import** setting the extrusion of layers can be switched on or off. The **Full 3D** option imports all layers with a thickness, as specified in the **Layers to import** table. With the **Metal shell** option layers are imported as embedded surfaces in one surface object (with grouping **All**), one surface object per layer (with grouping **By layer**), or one surface object per imported object (with grouping **No grouping**).

CONTROLLING THE LAYER STACKUP

Since layer stackup information is not included in GDS-II files, the layers are assigned a default zero thickness in the **Layers to import** table. These values can be edited before importing the GDS-II file.

Select the **Manual control of elevations** check box to position the layers in the *z* direction. This check box is enabled when **Grouping of geometries** is set to **By layer** or **No grouping**. When **Manual control of elevations** is not enabled, the *z* positions of the layers are calculated automatically from the layer **Thickness** values.

Select the **Show names from file** check box to display the layer names as included in the GDS-II file. This is useful after assigning new layer names that are used in the COMSOL application.

If the datatype record is used in the file for grouping of objects on a layer you can select the **Split by datatype** check box to treat geometry objects of the same datatype as a separate layer. The parent layer is then split into as many separate layers as there are datatypes defined. For example, a layer that contains objects that are grouped into two datatypes will be split into two separate layers after selecting the check-box. The name of the new layers is obtained by appending the datatype to the name of the parent layer.

Note: When the **Split by datatype** check box is selected the import will generate the geometry only for those layers that contain objects with the datatype record set in the file.

The **Layers to import** table contains the following columns:

- The **Name** column displays the layer names from the file. You can edit the entries to give more descriptive names to the layers. The names that appear here are used to name the resulting geometry objects and the selections when the **Layer selections** check box is selected under **Selections of Resulting Entities**.

- The **Name in file** column is visible only when you select the **Show names from file** check box above the table. The **Name in file** column always displays the layer name from the GDS-II file, even after editing the **Name** column.

- The **Type** column declares the type of layers. GDS-II files do not specify layer types, and the layer type is always set to **Metal**.

- The values in the **Thickness** column are used as the extrusion distances for the layers. Layer elevations are also calculated based on these values when **Manual control of elevations** is not enabled. The values in the layer **Thickness** column can always be changed prior to import.

- The values in the **Elevation** column control the lower z positions of the layers. By adjusting the values you can adjust the position of layers in the stackup. The **Elevation** column is only displayed when **Manual control of elevations** is enabled.

- The **Import** column. Clear the check box for layers that do not need to be imported.

To save the layers table to a text file, click the **Save to File** button (![icon]) under the table. To load layer table information from a text file, click the **Load from File** button (![icon]) under the table. Only the editable data in the table is affected when loading from file. Layer data in the file that does not match the existing layers in the table is ignored. To reset the thickness and elevation values in the table to the last read data from the GDS file click the **Reset Thickness and Elevation** button (![icon]). To select the **Import** check box in the table for all layers click the **Select All Metal and Dielectric Layers for Import** button (![icon]). Click the **Clear All Imports** button (![icon]) to clear all check boxes in the **Import** column. To reload the layer information from the GDS file to the table click the **Reload Layers** button (![icon]).

INTERIOR BOUNDARIES

By default the **Keep interior boundaries** check box is cleared to instruct the import to remove all interior boundaries on the imported layers. This keeps the geometry complexity to a minimum and can also make the import more robust in some situations.

ARC RECOGNITION

Recognition of arcs and straight lines can significantly reduce the complexity of imported layouts from GDS files. With the **Recognize arcs** set to **Automatic**, all polygon chains that represent arcs are identified and replaced with more efficient curve objects, and polygon segments that lie on a single straight line are recognized and joined into a single straight segment.

With **Recognize arcs** set to **Manual**, the following settings will help you fine tune the process of the merging of segments into a single circular arc:

- **Minimum angle between segments**: this parameter prohibits merging of two adjacent polygon segments to a circular arc if the angle between them is less than the provided number.

- **Maximum angle between segments**: this prohibits merging of two adjacent polygon segments to a circular arc if the angle between them is greater than the provided number.

- **Maximum curvature deviation**: this prohibits merging a polygon segment to the constructed circular arc if that would (relatively) change the curvature by more than the provided number.
- **Maximum length deviation**: this prohibits merging of two adjacent polygon segments to a circular arc if the relative difference between the lengths is greater than the provided number.
- **Maximum deviation from circle**: the allowed maximum relative error between the polygon segments and the constructed circular arc.

In manual arc recognition mode, the **Find straight lines** check box controls whether to convert several polygon segments that lie on a straight line into a single straight segment. This option uses the number in the **Minimum angle between segments** field to determine if a group of segments lie on the same straight line.

REPAIR

Geometry repair is controlled via the **Repair imported data** check box and the **Repair tolerance** list. Change the **Repair tolerance** from **Automatic** to **Relative** to manually specify a tolerance in the **Relative repair tolerance** edit field. The geometry repair allows to repair incorrectly drawn objects, such as polygons with (small) gaps.

If selected, the geometry repair is also active when combining the imported objects in a cell or layer. By increasing the repair tolerance, problems with short edges arising when combining the objects can sometimes be circumvented.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** check box to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These

selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Select the **Layer selections** check box to create predefined selections — for domains, boundaries, and objects — in subsequent nodes in the geometry sequence for each imported layer in the GDS file. To also make all of one of the resulting entities that the layers consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, or **Boundary selection**. The default is **All levels**, which makes predefined selections available on all applicable levels, suitable for use with materials and physics defined in domains and boundaries. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Select the **Cell selections** check box to create predefined selections — for domains and boundaries — in subsequent nodes in the geometry sequence for each imported cell in the GDS file. To also make all of one of the resulting entities that the cells consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, or **Boundary selection**. The default is **Domain selection**, if available, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Meshing an Imported Geometry

The imported geometry often consists of objects with very high aspect ratios, which are hard to mesh with a free tetrahedron mesh generator. As a result, it is often necessary to use interactive meshing of the imported geometry in a by-layer fashion.

The following section describes this procedure in general terms.

This procedure assumes that the top and bottom layers are metal layers. All metal layers can often be meshed using swept meshing, but dielectric layers usually cannot be meshed that way. Begin by meshing from the bottom or top layer, starting with a boundary mesh. Then mesh layer by layer, where each metal layer gets a swept mesh, and each dielectric layer (with vias) gets a free mesh.

The dielectric layers cannot use a swept mesh because the source and target boundaries usually do not look the same. If there is a surrounding air domain it is usually not possible to use swept meshes for the metal layers either. Use tetrahedra or convert the swept mesh to tetrahedra before meshing the surrounding domain.

Meshing and Convert in the *COMSOL Multiphysics Reference Manual*.

Troubleshooting ECAD Import

In this section:

- Tuning Import Settings
- Problems When Extruding Layers
- Problems with Several Geometry Objects

Tuning Import Settings

DELETE INTERIOR EDGES

A complex layout produces a large number of faces that can be hard to render. A simple way to reduce the number of faces is to clear the **Keep interior boundaries** check box in the ECAD import options. This removes all faces internal to the nets within a layer.

REMOVING FEATURES

Remove all features that are not important for the simulation. This is usually best to do before the import in the ECAD software. When importing with **Grouping of geometries** set to **No grouping** it is possible to manually delete certain objects after import, but it is recommended to do this only for relatively simple geometries.

Problems When Extruding Layers

Most ECAD or EDA programs support design rule checks (DRC), which test the entire layout and check that all features (vias, conductors, and components) are separated according to certain rules. With such checks the layout is free from overlapping vias and conductors touching other conductors or vias. This also ensures that the special extrude functionality of the ECAD import works properly. If the file contains such design-rule violations, the extrude operation might fail and issue an error message stating that it could not handle the topology of the layout.

The best approach to handle such problems is to perform a DRC with your ECAD software and produce new layout files. If this is not possible, import the layout in 2D and try to identify the problematic features.

Problems with Several Geometry Objects

If the special extrude functionality is not used, you get several geometry objects, for example, one for each layer if **By layer** is selected from the **Grouping of geometries** list. After a CAD import COMSOL Multiphysics is in the Geometry branch of the model tree. When you continue to the **Materials** branch if the model tree or to a physics interface node or the **Mesh** branch, the program tries to combine all the objects into one geometry, and this operation might fail if the objects are very complex and have high aspect rations. Resolve this by trying the option **All** in the **Grouping of geometries** list. This creates one combined geometry object by using the special extrude functionality, and with only one object this.

Another possibility is to use assemblies, because then COMSOL Multiphysics does not have to combine the objects (parts). This is controlled by the **Form Union/Assembly** node in the Geometry branch of the model tree. When using an assembly, use identity pairs to connect the interfaces between the layers.

As a final option, do not import the dielectric layers. The import then leaves isolated metal layers that have to connect with coupling variables.

3

Programming and Command Reference

In this section you find detailed COMSOL API reference information for the geometry features in the ECAD Import Module.

In this section:

- Commands Grouped by Function

Commands Grouped by Function

Commands for File Import

FUNCTION	PURPOSE
Import	Import geometry objects from a 3D CAD file.

Import

PURPOSE

Import geometry objects from an ECAD file in 2D and 3D using the ECAD Import Module

SYNTAX

```
model.geom(<tag>).feature().create(<ftag>,"Import");
model.geom(<tag>).feature(<ftag>).set(property,<value>);
model.geom(<tag>).feature(<ftag>).getType(property);
model.geom(<tag>).feature(<ftag>).getStringMatrix("layerprop");
model.geom(<tag>).feature(<ftag>).importData();
```

DESCRIPTION

Use `model.geom(<tag>).feature().create(<ftag>,"Import")` to create a geometry import feature.

When the property `filename` is set to a file recognized as an ECAD file, the property `ecadtype` is automatically initialized to either `gds`, `ipc2581`, or `odb++`. The following properties are available:

TABLE 3-1: VALID PROPERTY/VALUE PAIRS

PROPERTY	VALUE	DEFAULT	DESCRIPTION
ecadtype	gds | ipc2581 | odb++ | unknown	unknown	Type of ECAD file. A read only property that is automatically initialized from the property `filename`.
elevation	String[]		Z-position of layers.
filename	String		File name.
fileunit	String		The length unit in file. Read only.

TABLE 3-1: VALID PROPERTY/VALUE PAIRS

PROPERTY	VALUE	DEFAULT	DESCRIPTION
grouping	all | layer | none	layer	The grouping of the imported layers, where all returns one single object, and layer gives you one object per layer. In 2D, all is the same as layer.
height	String[]		Thickness of layers.
importlayer	String[]		The entries in the array govern if the corresponding layer is to be imported. Each entry value is either "on" or "off".
importtype	full3d | shell	full3d	Determine if metal layers are imported as solid or faces in 3D.
intbnd	on | off	off	Keep interior boundaries on layers.
layername	String[]		The user names for the layers. Empty string implies the name is taken from the ECAD file.
manualelevation	on | off	off	Manual control of elevations.
repairgeom	on | off	on	Repair imported objects.
repairtol	double	1e-5 (for GDS) 1e-6 (for ODB++ and IPC-2581	Repair tolerance, relative to size of union of imported objects. Setting this property will automatically set repairtoltype to relative and repairgeom to on.
repairtoltype	auto | relative	auto	Determine if repair tolerance is set automatically or manually by the repairtol property.
selresult	on | off	off	Create selections of all resulting objects.

TABLE 3-1: VALID PROPERTY/VALUE PAIRS

PROPERTY	VALUE	DEFAULT	DESCRIPTION
selresultshow	all | obj | dom | bnd | edg | pnt | off	The highest available entity level except obj; usually dom.	Show selections of resulting objects in physics, materials, and so on, or in part instances. obj is not available in a component's geometry. dom, bnd, and edg are not available in all features.
sellayer	on | off	on	Create layer selections.
sellayershow	all | dom | bnd | off	all	Show layer selections in physics, materials, and so on, and in part instances (when sellayer is on). This property is not available in a work plane's Plane Geometry.
updategeomunit	Boolean	true or false	The default is true if the feature is the first in the geometry sequence, else false. If true the geometry sequence unit will be updated to the fileunit value when performing the import.
contributeto	String	none	Tag of cumulative selection to contribute to.
createselection	on | off	off	Create selections.

The file specified by filename can be of any of the following formats:

TABLE 3-2: SUPPORTED FILE FORMATS

FILE FORMAT	FILE EXTENSIONS	ECADTYPE
GDS	.gds	gds
IPC-2581	.cvg,.xml	ipc2581
ODB++®	.zip, .tar, .tgz, .tar.tgz	odb++

The imported objects are represented using COMSOL's geometry modeler.

The property `ecadtype` determines which properties are available, see Table 3-2 for supported types. The property `ecadtype` is read only, and it is automatically initialized from the property `filename`, according to Table 3-2.

The property `importlayer` determines which layers to import. It is a string array of the same length as the number of layers, holding the strings `on` or `off`. The property is initialized with a valid default when setting the property `filename`.

To inspect available layers in a file, use the method

```
model.geom(<tag>).feature(<ftag>).getStringMatrix("layerprop");
```

It returns a `String[][]`, with the same number of rows as the number of layers, that contains information about each layer.

The property `height` determines the height of all layers. It is a string array of the same length as the number of layers, holding string representations of the layer height, for example `1[mm]`. The property is initialized with a valid default when setting the property `filename`.

The property `manualelevation` controls how imported layers are positioned in the Z direction. When `manualelevation` is `off`, the imported layers are stacked on top of each other with no gaps, so the Z positions are computed from the `height` property. When `manualelevation` is `on` and grouping is set to `layer` or `none`, the imported layers are positioned in the Z direction as specified by the `elevation` property. To switch to manual control of elevations it is recommended to first set `manualelevation` to `on`, then set the values for the `height` and `elevation` properties.

The property `elevation` determines the Z-position of all layers. It is a string array of the same length as the number of layers, holding string representations of the layer elevation. The property is initialized with a valid default on these occasions:

- when setting the property `filename`,
- when setting the property `manualelevation` to `off`,
- when `manualelevation` is `off`, and the value of the `height` property is changed,
- when `manualelevation` is `off`, and the value of the `importtype` property is changed.

If `selresult` is set to `on`, a selection is created for all resulting entities of each type (object, domain, boundary, edge, and point), for use in the geometry sequence. To access the object selection, use `model.geom(<tag>).selection(<ftag>)`, where `<tag>` is the geometry tag and `<ftag>` is the feature tag. To access the other selections,

use `model.geom(`*`<tag>`*`).selection(`*`<ftag>.<lvl>`*`)`, where *`<tag>`* is the geometry tag, *`<ftag>`* is the feature tag, and *`<lvl>`* is one of `dom`, `bnd`, `edg`, or `pnt`. If, in addition, `selresultshow` is set to a value other than `off`, all or some of these selections appear for use outside the geometry sequence. To access these selections, use `model.selection(`*`<tag>_<ftag>_<lvl>`*`)`, where *`<tag>`* is the geometry tag, *`<ftag>`* is the feature tag, and *`<lvl>`* is one of `dom`, `bnd`, `edg`, or `pnt`.

If `sellayer` is set to `on`, a selection is created for all resulting entities of the types object, domain, and boundary, of each layer, for use in the geometry sequence. To access the object selections, use `model.geom(`*`<tag>`*`).selection(`*`<ftag>_<otag>`*`)`, where *`<otag>`* is a tag derived from the name of the imported layer. *`<otag>`* is derived by replacing space and dot characters with underscore characters and removing other characters that are not numbers or uppercase or lowercase English characters (A–Z and a–z). Additionally, if required to make *`<otag>`* unique, `_`*`<m>`* is appended, where *`<m>`* is an integer. To access the other selections, use `model.geom(`*`<tag>`*`).selection(`*`<ftag>_<otag>`*`.dom`) or `model.geom(`*`<tag>`*`).selection(`*`<ftag>_<otag>`*`.bnd`), where *`<otag>`* is a tag derived from the name of the imported layer. If, in addition, `sellayershow` is set to a value other than `off`, all or some of these selections appear for use outside the geometry sequence.

To access these selections, use `model.selection(`*`<tag>_<ftag>_<otag>`*`_dom`) or `model.selection(`*`<tag>_<ftag>_<otag>`*`_bnd`), where *`<otag>`* is a tag derived from the name of the imported layer.

If the file type is GDS, you can create additional selections for each imported cell by using the properties `selcell` and `selcellshow`.

Properties that are only available when `ecadtype` is `gds`.

TABLE 3-3: VALID PROPERTY/VALUE PAIRS FOR GDS

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
arcdistancetol	double	0.3	Tolerance for segment length deviation.
arcmaxangle	double	36	Maximum angle for segments.
arcminangle	double	3.6	Minimum angle for segments to part of an arc.
arcmaxcircledev	double	1e-3	Maximum allowed point deviation from a circle, relative to the circle radius.

TABLE 3-3: VALID PROPERTY/VALUE PAIRS FOR GDS

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
arcradiustol	double	0.4	Tolerance for arc curvature.
cell	String	empty	Name of cell to import. The top cell is imported by default.
cellfilter	String	empty	Name of cell to include. Leave empty to include all cells below the cell selected by the cell property.
findarcs	auto | manual | off	auto	Control arc recognition.
findlines	on | off	on	Turn on straight line recognition. Only available when findarcs is manual.
selcell	on | off	off	Create cell selections.
selcellshow	all | dom | bnd | off	dom	Show cell selections in physics, materials, and so on, and in part instances (when selcell is on). This property is not available in a work plane's Plane Geometry.
splitbydatatype	on | off	off	Treat datatypes as separate layers

When findarcs is auto or manual, line segments are combined to form arcs. If findarcs is manual, the properties arcdistancetol, arcradiustol, arcminangle, arcmaxangle, and findlines can be used to tune the arc recognition algorithm, otherwise the algorithm tries to determine optimal parameters.

Properties that are only available when ecadtype is ipc2581, odb++.

TABLE 3-4: VALID PROPERTY/VALUE PAIRS FOR IPC-2581 AND ODB++

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
dielectric	on | off	on	Import dielectric regions
ignorelinewidth	on | off	off	Import line symbols as curve objects by ignoring the line width during import
ignoreoffset	double	-0.1[mm]	Offset used when ignoreoutsideboard is on. The file unit is used if no unit is specified.

TABLE 3-4: VALID PROPERTY/VALUE PAIRS FOR IPC-2581 AND ODB++

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
`ignoreoutsideboard`	`on	off`	`off`	Ignore objects for which the defining points fall outside the board boundary incremented with the offset defined by `ignoreoffset`
`ignoretext`	`on	off`	`on`	Ignore objects marked as text in the files
`selnetkeepbnd`	`on	off`	empty	Keep boundary net selections
`selnetkeepdom`	`on	off`	empty	Keep domain net selections
`selnetnamebnd`	`String[]`	empty	Names of boundary net selections (read-only)
`selnetnamedom`	`String[]`	empty	Names of domain net selections (read-only)
`selnettagbnd`	`String[]`	empty	Tags of boundary net selections (read-only, hidden in GUI)
`selnettagdom`	`String[]`	empty	Tags of domain net selections (read-only, hidden in GUI)
`selnetcontributeto bnd`	`String[]`	empty	Tags of cumulative selection to contribute to (or none to not contribute), for boundary net selections
`selnetcontributeto dom`	`String[]`	empty	Tags of cumulative selection to contribute to (or none to not contribute), for domain net selections
`selnetshowbnd`	`on	off`	empty	Show boundary net selections in physics, materials, and so on; in part instances; or in 3D from a plane geometry
`selnetshowdom`	`on	off`	empty	Show domain net selections in physics, materials, and so on; in part instances; or in 3D from a plane geometry

You can use the net selection tag as a key when setting array elements in the other properties by using `set(property,<ntag>, value)`, where *<ntag>* is a tag derived from the name of the net selection.

To access net selections, use `model.selection(`*`<tag>_<ftag>_<ntag>`*`_dom`) or `model.selection(`*`<tag>_<ftag>_<ntag>`*`_bnd`), where *`<ntag>`* is the tag of the net selection.